DESIGN OPTIMIZATION OF CONNECTING ROD FOR INTERNAL COMBUSTION ENGINE

MOHD SHAMIL BIN SHAARI

Report submitted in partial of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled “Design Optimization of Connecting Rod for Internal Combustion Engine” is written by Mohd Shamil Bin Shaari. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

(ENGR. ZAMRI MOHAMED)

Examiner   Signature

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this report and in my opinion this report is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

Signature

Name of Supervisor : DR. MD. MUSTAFIZUR RAHMAN

Position : ASSOCIATE PROFESSOR

Date : 6 DECEMBER 2010

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted in candidate of any other degree.

Signature

Name : MOHD SHAMIL BIN SHAARI

ID Number : MH08086

Date : 06 DECEMBER 2010

ACKNOWLEDGEMENT

In the name of Allah, the Most Benevolent, the Most Merciful. Alhamdulillah, all praises to Allah, the Almighty, on whom ultimately we depend for sustenance and guidance. All praises to Allah for the strengths and His blessing in completing this report.

Special appreciation goes to my supervisor, Associate Professor Dr. Md. Mustafizur Rahman, whose guidance, careful reading and constructive comments was valuable. His timely and efficient contribution helped me shape this into its final form and I express my sincerest appreciation for his assistance in any way that I may have asked. His invaluable help of constructive comments and suggestions throughout the report have contributed to the success of this project. My sincere thanks also go to Ir. Zamri Mohamed, I owe special thanks for his consultations especially on this report. Certainly, not forgetting the UMP Faculty of Mechanical Engineering for providing the support and equipment required in order to completing this study.

Sincere thanks to all my friends especially Alliff, Azlan, Raja Shahrul, Omar, Zulfadli, Ahmad Nazmi, Ezuwanizam, ‘Azim, Hishammudin, Azam and others for their kindness and moral support during my study. Thanks for the friendship and memories.

Last but not least, my deepest gratitude goes to my beloved parents; Mr. Shaari Bin Sudin and Mrs. Roshadah Binti Hamid and to my brothers for their endless love, prayers and encouragement. To those who indirectly contributed in this research, your kindness means a lot to me. Thank you very much.

ABSTRACT

This project describes the study of topology optimization for connecting rod of internal combustion engine. The objectives of this study are to develop structural modelling, finite element analysis and the optimization of the connecting rod for robust design using topology optimization technique. The structure of connecting rod was modelled using SOLIDWORKS software. Finite element modelling and analysis were performed using MSC.PATRAN and MSC.NASTRAN software. Linear static analysis was carried out to obtain the stress/strain state results. The mesh convergence analysis was performed to select the best mesh for the analysis. The topology optimization technique is used to reduce the weight of the connecting rod. From the FEA analysis results, TET10 predicted higher maximum stress than TET4 and maximum principal stress captured the maximum stress. From the topology optimization, the crank end was suggested to be redesign. The optimized connecting rod is 11.7% lighter and predicted lower maximum stress compare to initial design. For future research, the optimization should cover on material optimization to increase the strength of the connecting rod.

ABSTRAK

Projek ini menggambarkan kajian pengoptimuman topologi untuk batang penghubung enjin pembakaran dalaman. Tujuan kajian ini adalah untuk membangunkan model struktur, analisis unsur terhingga dan optimalisasi batang penghubung menggunakan teknik pengoptimuman topologi. Struktur batang penghubung dimodelkan menggunakan perisian SOLIDWORKS. Pemodelan dan analisis unsur terhingga dilakukan dengan menggunakan perisian MSC.PATRAN dan MSC.NASTRAN. Analisis linear statik dilakukan untuk mendapatkan hasil tekanan/regangan. Analisis konvergensi jejaring dilakukan untuk memilih jejaring terbaik untuk analisis. Teknik optimasi topologi digunakan untuk mencapai tujuan dari pengoptimuman bagi mengurangkan berat batang penghubung. Dari hasil analisis FEA, TET10 menganggarkan tekanan maksimum lebih tinggi dari TET4 dan tekanan utama maksimum dapat menangkap bacaan tertinggi. Berdasarkan hasil pengoptimuman topologi, akhir engkol disarankan untuk direka bentuk semula. Batang penghubung dapat dioptimumkan 11.7% lebih ringan dan dijangka tekanan maksimum yang rendah berbanding reka bentuk awal. Untuk kajian akan datang, pengoptimuman harus merangkumi pengoptimalan bahan untuk meningkatkan kekuatan batang penghubung.

TABLE OF CONTENTS

		Page
SUPERVISOR'S DECLARATION		ii
STUDENT'S DECLARATION		iii
ACKNOWLEDGEMENTS		iv
DEDICATION		v
ABSTRACT		vi
ABSTRAK		vii
TABLE OF CONTENTS		viii
LIST OF TABLES		x
LIST OF FIGURES		xi
LIST OF SYMBOLS		xiii
LIST OF ABBREVIATIONS		xiv
CHAPTER 1	**INTRODUCTION**	
1.1	Background	1
1.2	Problem Statements	2
1.3	Objectives of Project	2
1.4	Scope of study	3
1.5	Outline of Report	3
CHAPTER 2	**LITERATURE REVIEW**	
2.1	Introduction	4
2.2	Connecting Rod	4
2.3	Finite Element Modelling and Analysis	5
2.4	Optimization of Connecting Rod	7
2.5	Conclusion	9

CHAPTER 3 METHODOLOGY

3.1 Introduction 10
3.2 Theoretical Basis of Connecting Rod 10
3.3 Project Flowchart 12
3.4 Modelling of The Connecting Rod 13
3.5 Linear Static Analysis 14
3.6 Optimization Technique 15
3.7 Conclusion 19

CHAPTER 4 RESULTS AND DISCUSSION

4.1 Introduction 20
4.2 Material Information 20
4.3 Finite Element Modelling 21
4.4 Identification of The Mesh Convergence 29
4.5 Modal Analysis 30
4.6 Optimization of Connecting Rod 32
4.7 Proposed New Design 36
4.8 Conclusion 42

CHAPTER 5 CONCLUSION AND RECOMMENDATION

5.1 Introduction 43
5.2 Conclusion 43
5.3 Recommendation 44

REFERENCES 45

APPENDICES

A Engineering Drawing of Connecting Rod 47

LIST OF TABLES

Table No.	Title	Page
4.1	Mechanical properties of C-70 Steel	20
4.2	Variation of stresses concentration at the critical location of the connecting rod for TET4 mesh	27
4.3	Variation of stresses concentration at the critical location of the connecting rod for TET10 mesh	27
4.4	Variation of mesh size related to number of element and node for TET10	29
4.5	The result of modal analysis	30
4.6	Objective function and maximum constraint history from optimization of connecting rod	33
4.7	Comparison between initial and optimized designs on stress and mass	39

LIST OF FIGURES

Figure No.	Title	Page
2.1	Finite element mesh of connecting rod pin end	6
2.2	Optimized connecting rod	7
3.1	Coordinate system of connecting rod	11
3.2	Optimization based on finite element analysis	12
3.3	Isometric view of 3D modelling of connecting rod	13
3.4	3D modelling of initial connecting rod	13
3.5	Relationship between load and displacement	14
3.6	Flowchart to perform the linear static analysis	15
3.7	Flowchart of optimization approach	16
3.8	Power-law approach	17
4.1	Finite element model with difference mesh size	23
4.2	FEM using 8 mm mesh size with nodes and element	24
4.3	Boundary condition with tensile load	25
4.4	Von-Mises stresses contour	26
4.5	Comparison between TET4 and TET10 on maximum displacement	27
4.6	Comparison of stresses betweenTET4 and TET10	28
4.7	Stresses concentration versus mesh global lenght for TET10 to check mesh convergence	29
4.8	The mode shapes of the connecting rod	31
4.9	Optimized connecting rod	34
4.10	Objective function history	35
4.11	Maximum constraint history	35

4.12	Isometric 3D view of new design	37
4.13	Different view of optimized design	38
4.14	3D view of optimized design superimposed with initial design	39
4.15	Top 3D view of optimized design superimposed with initial design	39
4.16	Front 3D view of optimized design superimposed with initial design	40
4.17	Maximum principle stresses contour	41

LIST OF SYMBOLS

P	Normal pressure
P_o	Normal pressure constant
P_t	Total resultant load (Tensile)
P_c	Total resultant load (Compression)
E	Young's modulus
$\mathbf{K}$	Stiffness matrix
$\mathbf{u}$	Vector of displacements
$\mathbf{f}$	Vector of applied forces
x	Relative density
ρ	Density
c	Compliance
x	Stiffness
g_e	Design variable

LIST OF ABBREVIATIONS

BC	Boundary Condition
CAD	Computer-aided design
CAE	Computer-aided engineering
CAO	Computer Aided Optimization
DOF	Degree of freedom
FE	Finite element
FEA	Finite Element Analysis
FEM	Finite element modelling
HEXA	Hexahedral
IC	Internal Combustion
PENTA	Pentahedral
RE	Reverse engineering
TET	Tetrahedral

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Connecting rod is a high volume production critical component that being use in automotive internal combustion engine. It connects the piston to rotating crankshaft and transmitting the power of combustion through piston then to the crankshaft. Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. They can be produced either by casting, powder metallurgy or forging (Grass et al., 2006). However, connecting rods could be produced by casting and the difference between others it usually have blow holes which are adverse from durability and fatigue points of view. The fact that forgings produce blow hole-free and better rods gives them an advantage over cast rods (Gupta, 1993). Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste. However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques (Repgen, 1998).

Reverse engineering (RE) is the process of taking something like a device, a mechanical component or even a software apart and analyzing its workings in detail, usually with the intention to construct a new device or program that does the same thing without actually copying anything from the original (Hashmi, 2005). In order to boost the quality and efficiency of the design, RE should be used. It is not just redesign the product but also to make sure that the new component can achieve more than original product.

1.2 PROBLEM STATEMENTS

Connecting rod is one of the most critical components internal combustion (IC) engines bearing the statically and dynamically fluctuating loads (Hashmi, 2005). The optimization of connecting rod had already started as early year (Webster, 1983). However, everyday consumers are looking for the best from the best. That“s why the optimization is really important for automotive industry especially. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less total cost productions. The design and weight of the connecting rod influence on car performance. Hence, it effects on the car manufacture credibility. Change in the design and material results a significant increment in weight and also performance of the engine. The structural factors considered for weight reduction during the optimization include the buckling load factor, stresses under the loads, bending stiffness, and axial stiffness. Thus, the component can give the higher strength, efficient design and lighter that would create a major success in the automotive and manufacturing industry. The benefits of connecting rod optimization are eventually go back to consumer itself. Among the main objectives are to improves the engine performance and also to strengthen the product that is ensure the safety of human being. It undergoes high cyclic loads of the order of 10^8”10^9 cycles, which range from high compressive loads because of combustion, to high tensile loads because of inertia (Shenoy, 2004). Connecting rod failed due to insufficient strength to hold the load. By maximize the strength, automatically it will longer the life cycles of the connecting rod. Lots of knowledge will be apply and produced during the process. In this study, the design of the connecting rod will be improve and at the same time increase the strength. The study will be focus on the finite element modeling and analysis. From the analysis results, the decision whether connecting rod needs to be redesign or not will based on the topology optimization.

1.3 OBJECTIVES OF PROJECT

The objectives of the project are as follows

(i) To develop structural modeling of connecting rod

(ii) To perform finite element analysis of connecting rod

(iii) To develop structural optimization model of connecting rod

1.4 SCOPE OF PROJECT

Before performing the topology optimization, the structural modeling of the connecting rod needs to be developed by using computer-aided design (CAD) software. The structural modeling then imported into the computer-aided engineering (CAE) and began the meshing on the connecting rod. The finite element modeling (FEM) processes were performed by using MSC.PATRAN. The boundary condition (BC) and loading selected and place at the connecting rod. The finite element analysis (FEA) then carried out at the connecting rod. The MSC.NASTRAN used to solve the analysis equation from MSC.PATRAN. Thus, producing the result of stress, strain and displacement where it will be used to analyze the critical area of the connecting rod. Finally the topology optimization will take place and the result will be used to design new connecting rod.

1.5 OUTLINE OF REPORT

Chapter 1 introduces the background, problem statement and the scopes of this study. Chapter 2 presents the literature study about connecting rod, finite element method and optimization of the connecting rod. Chapter 3 discusses the development of structural modeling, finite element modeling and the optimization technique. Chapter 4 discusses the results and analysis of the finite element analysis, modal analysis and optimization of the connecting rod. Chapter 5 presents the conclusion and recommendation of the future work.

CHAPTER 2

LITERITURE REVIEW

2.1 INTRODUCTION

The purpose of this chapter is to provide information which related to the connecting rod, finite element analysis (FEA) and also about optimization of connecting rod.

2.2 CONNECTING ROD

In modern automotive internal combustion engines, the connecting rods are most usually made to absorb high impact stresses that occur onto it. Rasekh et al. (2009) explained about study of experimental equation that was performed for a Tractor MF-285 connecting rod and also using FEA. The maximum stresses in different parts of MF-285 connecting rod were determined. From the analysis, three parts were being considered of the stress distributions which are pin end, rod and crank end. Finally, a comparison between FEA results and experimental equation method were made.

Mirehei et al. (2008) investigated the connecting rod fatigue of universal tractor (U650) was through the ANSYS software application and its lifespan was estimated. The connecting rod behavior affected by fatigue phenomenon due to the cyclic loadings and to consider the results for more savings in time and costs, as two very significant parameters relevant to manufacturing. The results indicate that with fully reverse loading, one can estimate longevity of a connecting rod and also find the critical points that more possibly the crack growth initiate from. It is suggested that the results obtained can be useful for modifications in the process of connecting rod.

Afzal and Fatemi (2004) investigate and compare fatigue behavior of forged steel and powder metal connecting rods. The experiments included strain-controlled specimen testing, with specimens obtained from the connecting rods, as well as load-controlled connecting rod bench testing. Monotonic and cyclic deformation behaviors, as well as strain-controlled fatigue properties of the two materials are evaluated and compared. Experimental S-N curves of the two connecting rods from the bench tests obtained under R = -1.25 constant amplitude loading conditions are also evaluated and compared. Fatigue properties obtained from specimen testing are then used in life predictions of the connecting rods, using the S-N approach. The predicted lives are compared with bench test results and include the effects of stress concentration, surface finish, and mean stress. The stress concentration factors were obtained from FEA was used to account for the mean stress effect. Fractography of the connecting fracture surfaces were also conducted to investigate the failure mechanisms. A discussion of manufacturing cost comparison and recent developments in 'crackable' forged steel connecting rods are also included.

2.3 FINITE ELEMENT MODELING AND ANALYSIS

The objective of FEA was to investigate stresses and hotspots experienced by the connecting rod. From the resulting stress contours, the state of stress as well as stress concentration factors can be obtained and consequently used for life predictions (Afzal and Fatemi, 2004).

Rahman (2009) discuss about FEA of the cylinder block of the free piston engine. The 4 nodes tetrahedral (TET4) element version of the cylinder block was used for the initial analysis. The comparison then are made between the TET4 and the 10 nodes tetrahedral (TET10) element mesh while using the same global mesh length for the highest loading conditions (7.0 MPa) in the combustion chamber. From the results, the TET10 mesh predicted higher von Mises stresses than that the TET4 mesh. The TET10 mesh is presumed to represent a more accurate than TET4 meshes. TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function. For the same element size, the TET10 is expected to be able to capture the high stress concentration associated with the bolt holes. A TET10 was then finally used for the solid mesh. Mesh study is performed on FE model to ensure sufficiently fine sizes are employed for accuracy of calculated results but at competitive cost (CPU time).

Bari et al. (2004) compare FEA of slab with others analytical solution. Slabs are most widely used structural elements that transmit load to the supporting walls and beams and sometimes directly to the columns by shear and torsion. Similarly with various classical mathematical procedures, simple beams were analyzed in which the concrete and the steel reinforcement were represented by two-dimensional triangular finite elements. Special bond link elements were used to connect the steel to the concrete. Linear analysis were performed on beams with predefined crack patterns to determine principal stresses in the concrete, stresses in the steel reinforcement and bond stresses. From the observations of the results, displacements and moments by using FEM were 20% more accurate than the analytical results. Accurate results for moments may be obtained if more refined mesh is taken into accounts.

Yang et al. (1992) were meshing finite element modeling. The connecting rod pin end as shown in Figure 2.1 contains 1012 linear HEXA elements, 4 linear PENTA elements, and 1569 grid points. The design goal is to minimize the material volume subject to a constraint on the von Mises stress. This constraint is imposed at each node in the finite element model of the connecting rod head except the nodes at the reentrant corner where the wrist pin leaves the rod. The singularity effects that occur here can be considered by imposing a stress concentration factor, but the interface between the pin end and crank end generally requires complex modeling techniques.

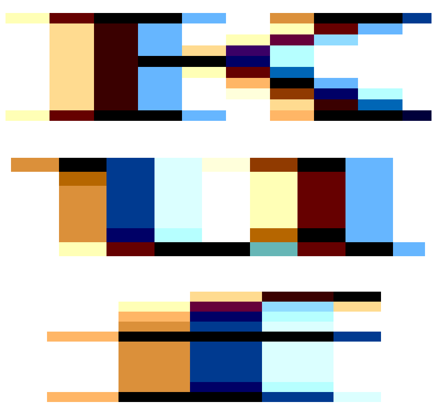

Figure 2.1: Finite element mesh of connecting rod pin end by (Yang et al., 1992).

2.4 OPTIMIZATION OF CONNECTING ROD

A connecting rod is subjected to many millions of repetitive cyclic loadings. So, typically designed for infinite-life and the primary design criterion is endurance limit. Therefore, the durability of this component is of critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance simulation, fatigue and a lot more. It is necessary to investigate finite element modeling techniques, optimization techniques and new design in reducing the weight and at the same time increase the strength of the connecting rod itself. Shenoy (2004) performed a study to explore weight and cost reduction opportunities for a production forged steel connecting rod. The study has dealt with two parts, firstly dynamic load and quasi-dynamic stress analysis of the connecting rod and secondly to optimize the weight and cost. From the results, the existing connecting rod can be replaced with a new connecting rod made of C-70 steel that is 0% lighter and 2% less expensive due to the steel“s fracture crackability. Yet, the same performance can be expected in terms of component durability without additional machining of the mating surfaces. The optimize design is show in Figure 2.2.

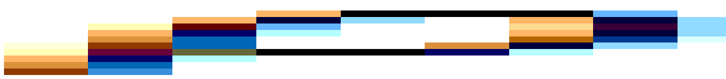

Figure 2.2: Optimized connecting rod by (Shenoy, 2004).

Zoroufi (2004) explained about exploring the design criteria and optimization potentials which is vital to the automotive industry. The study was aimed to developing general procedures for fatigue analysis and optimization of safety critical automotive components with manufacturing considerations. From the results, in terms of structural performance and durability based on both material testing and component evaluation, forged steel was found superior to cast iron which in turn was found superior to cast aluminum. In terms of overall weight and cost reductions of at least 12% and 5%, respectively, are estimated for the example part following the optimization task. The cost of the saved material is additional reduction, though not very considerable due to small portion of material cost within the total production cost.

Ulatowska (2008) explained about shape optimization of a connecting rod for a steel forged connecting rod by using computer aided optimization (CAO). The task purpose was reduction of large notch stresses. In initial model, there are stresses less than 100 MPa and more than 320 MPa, in during the optimization procedure has been try to keep Min and Max von Mises stress between 270 MPa and 300 MPa. When there is stress less than 270 MPa, CAO program remove material in order to increase the stress in this point. When there is more stress than 300 MPa, CAO add material in order to decrease the stress in this point.

Shenoy and Fatemi (2005) explained about optimization study was performed on a steel forged connecting rod with a consideration for improvement in weight and production cost. Weight reduction was achieved by using an iterative procedure. In this study weight optimization is performed under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads. The study results in an optimized connecting rod that is 10% lighter and 25% less expensive, as compared to the existing connecting rod.

Yang et al. (1992) describes a successful process for performing component shape optimization should be focused on design modeling issues. A modular software system is described and some of the modules are widely available commercial programs such as PDA/PATRAN and MSC/NASTRAN. The upper end (pin end) of an automotive connecting rod is optimized under a variety of initial assumptions to illustrate the use of the system.

2.5 CONCLUSION

This chapter has been the summary of previous works that related to this project. The works were discussed are about connecting rod, FEA and optimization of connecting rod. The next chapter will be discussed about the methodology of this project.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter presents the overall methodology of the optimization based on finite element analysis. The optimization is the most critical process in the automotive industry. It is very important that any production company invested millions of their profits into Research and Development (R&D). The aim of this chapter is to develop a methodology to improve the process of optimizing a certain product.

3.2 THEORETICAL BASIS OF CONNECTING ROD

Shenoy (2004) performed the optimization on the connecting rod for static FEA loading from Webster et al. (1983) experimental results. Figure 3.1 shows the coordinate system of connecting rod at crank end.

The normal pressure (P) on the contact surface is given by:

$$P = P_o \cos\theta \qquad (3.1)$$

The load is distributed over an angle of 180^o. The total resultant load (P_t) is given by:

$$P_t = \int_{-\frac{\pi}{2}}^{\frac{\pi}{2}} \left(P_o \cos\theta\right) r t \, d\theta = P_o r t \left(\frac{\pi}{2}\right) \qquad (3.2)$$